VIII International Conference on Computational Methods for Coupled Problems in Science and Engineering
COUPLED PROBLEMS 2019
E. Oñate, M. Papadrakakis and B. Schrefler (Eds)

NUMERICAL INVESTIGATION ON ANTI-ICING PERFORMANCE OF HEATING SURFACE FOR NACA0012 AIRFOIL

K. FUKUDOME*, S. URANAI*, H. MAMORI† AND M. YAMAMOTO*

* Department of Mechanical Engineering, Faculty of Engineering
Tokyo University of Science
6-3-1 Niijuku, Katsushika-ku, Tokyo 125-8585, Japan
e-mail: kfukudome@rs.tus.ac.jp, www.rs.kagu.tus.ac.jp/yamamoto/

† Graduate School of Informatics and Engineering, School of Informatics and Engineering
The University of Electro-Communications
1-5-1, Chofugaoka, Chofu, Tokyo 182-8585, Japan
email: mamori@uec.ac.jp, http://www.mamorilab.mi.uec.ac.jp/

Key words: Ice Accretion, Anti-Icing Method, Airfoil, Heating Surface, Super-Cooled Water Droplet.

Abstract. Ice accretion is a phenomenon that super-cooled water droplets impinge and accrete on wall surfaces. It is known that icing can cause severe accidents. To prevent the icing, an electro-thermal heater is recently adopted as the de- and anti-icing device for wings. In the present study, we conducted icing simulations of a two-dimensional NACA0012 airfoil with an electro-thermal heater on the leading-edge surface to optimize the heating area. The attack angle and the heating area were changed from 0 to 4 degrees and from 0 to 2.0% chord length, respectively. Through the simulations, we found that the lift coefficient was significantly improved by the heating, the drag coefficient generally decreased with increasing the heating area, and at the attack angle of 0 degree and the heating area of 1.0% chord length, the drag coefficient exceptionally became worse because of the residual ice shape with horns.

1 INTRODUCTION

Icing is a phenomenon that super-cooled water droplets collide on a solid surface and it forms an ice layer on the surface. The icing problem occurs for many industrial apparatuses such as power lines, buildings, cars, ships, aircraft wings, jet engines and so on. For aircraft wings, the icing makes ice layer and roughness on the surface and they lead a drastic decrease of aerodynamic performance. Therefore, the icing is one of the reasons to cause a fatal accident of aircraft. To prevent the accident, the prediction of icing and the development of de- and anti-icing techniques are very important from the viewpoints of aircraft safety. As the de- and anti-icing techniques, bleed air, anti-freezing liquid and an electric heater have been proposed and implemented for aircraft. Because of the easy setting and the low environmental burden, the electric heater has recently been adopted in current aircraft. We can find a number of technical papers on icing in the literature. For example, Al-Khalil et al. [1] performed experimental and numerical studies of icing on the wing with the electric heater. They reported the effect of the different heating temperatures of the electric heater. Bu et al. [2] conducted the icing simulation of a wing with considering heat flux from the wing surface. Reid et al. [3] performed the numerical simulation of an electric-thermal de-icing system, and they reported the temporal variation of surface temperature for the de-icing process. However, the research focusing on the heating area of the electric heater has not been conducted yet.

Taking into account these backgrounds, in the present study, we perform two-dimensional icing simulations of NACA0012 airfoil to investigate the effect of the heating area on the aerodynamic performance. Through this study, the influences of heating area on the residual ice formation, the drag and the lift of the airfoil and energy consumption are clarified.

2 NUMERICAL PROCEDURE

The numerical simulation consists of four steps: (1) generation of computational grids; (2) computation of flow field; (3) computation of droplet trajectories and collection efficiency; (4) thermodynamics computation. According to them, we can obtain ice shape on the airfoil. In the following, the details of each numerical procedure are briefly explained.

The computational target is a NACA0012 airfoil. The computational grid is shown in Fig. 1. The overlap grid is employed. The main is used to simulate the whole flow field around the airfoil, and as shown in Fig. 1, the sub grid has high resolution to correctly obtain the ice shape around the leading edge and the boundary layer on the airfoil. The total number of grid points is about 26,000. The convergence of the ice shape by the grid points has preliminarily been confirmed through the grid independent study.

The flow around the airfoil is assumed to be two-dimensional, compressible, and fully turbulent. The governing equations are the continuity, Navier-Stokes, energy equations and transport equations of the turbulent kinetic energy k and its dissipation rate ε. The Kato-Launder k-ε turbulent model [4] is employed to suppress the over-production of turbulence around the leading edge region.

The droplet trajectory is computed based on the Lagrangian approach. Since the size of droplet and the concentration are small enough, a one-way coupling method is employed. That is, the droplet motion is affected from the flow field, whereas the droplet does not affect the flow field. The motion equation of a droplet is a simplified BBO equation as

$$\frac{d\overrightarrow{U_d}}{dt} = \frac{3}{4} C_D \frac{\rho_g}{\rho_d} \frac{1}{d_d} \overrightarrow{U_r} \left| \overrightarrow{U_r} \right| \, . \tag{1}$$

Here, $\overrightarrow{U_d}$ is the velocity of a droplet and $\overrightarrow{U_r}$ is the relative velocity between the flow and the droplet. ρ_g and ρ_d are the density of the gas (i.e. air) and the droplet respectively, d_d is the diameter of the droplet, and C_D is the drag coefficient. Since we suppose that the droplet does not deform and rotate, as the drag coefficient of a sphere, Schiler model [5] is used, which is defined as

$$C_D = \frac{24}{\mathrm{Re}} \left(1 + 0.15 \mathrm{Re}^{0.687} \right), \tag{2}$$

where Re is the droplet Reynolds number based on the droplet diameter, the relative velocity, density of the fluid ρ_f, and viscosity of the fluid μ_f .

In the thermodynamics computation, since the time scales of the flow field and the icing are significantly different, we use the weak coupling method. The Extended Messinger Model [6] (referred as EMM, hereafter) is adopted as the icing model. The EMM is based on the mass and energy conservation law of the ice and water and the equation for phase change at the interface between the ice and the water. Although the EMM is widely used in icing simulations, the surface heating and the temperature of run-back water are not considered. Thus, in the present study, we improve the EMM to reproduce them as follows. First, the heat flux is computed by the temperature of the colliding droplet. Second, the temperature and thickness of the ice or water film are computed. If the temperature is over 0 degree Celsius, the adhered droplet is treated as a water film.

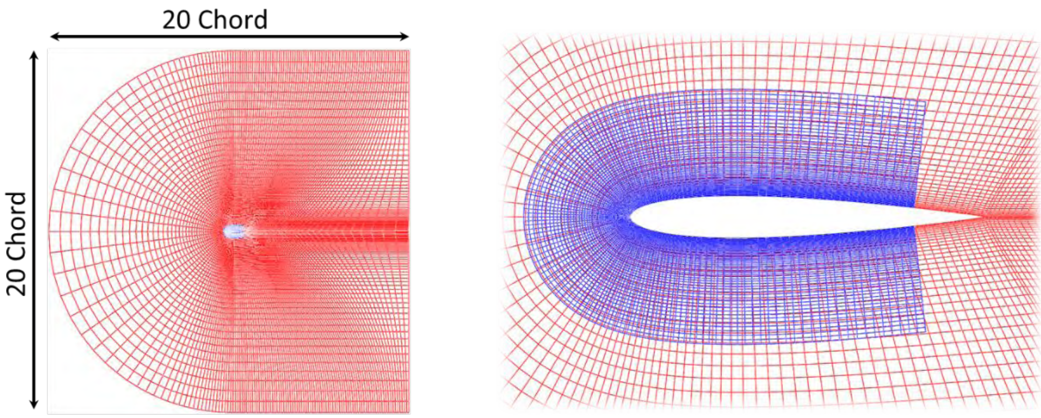

Figure 1 Computational domain and grid. Left and right figures represent the entire computational domain and the enlarged view of sub-grid around airfoil, respectively. Main and sub grids are colored in red and blue, respectively.

The computational conditions are tabulated in Table 1, which are set in accordance with the experimental study performed by Olsen et al. [6]. In the present study, the anti-icing simulation of the NACA airfoil is carried out and therefore the surface of the airfoil is partially heated. At the heated region, the constant temperature is fixed, and at the other region, the adiabatic condition is imposed. The three different angle of attack and heating-region cases are examined as tabulated in Table 2. The heater temperature is set to be 5.0 ºC for all cases with heating.

Table 1: Computational conditions

Static Temperature	[ºC]	-27.8
Accretion Time	[sec.]	480
Inflow Velocity	[m/s]	58.1
MVD (Median Volume Diameter)	[μm]	20.0
LWC (Liquid Water Content)	[g/m^3]	1.3
Chord Length	[m]	0.53
Attack Angle	[deg.]	0.0
Ambient Pressure	[kPa]	95.6
Exposure Time	[s]	480

Table 2: Parameters of test cases

Case	A00	A01	A02	A20	A21	A22	A40	A41	A42
Attack Angle [deg]		0.0			2.0			4.0	
Heating Area *x*/*c* [%]	0.0	1.0	2.0	0.0	1.0	2.0	0.0	1.0	2.0

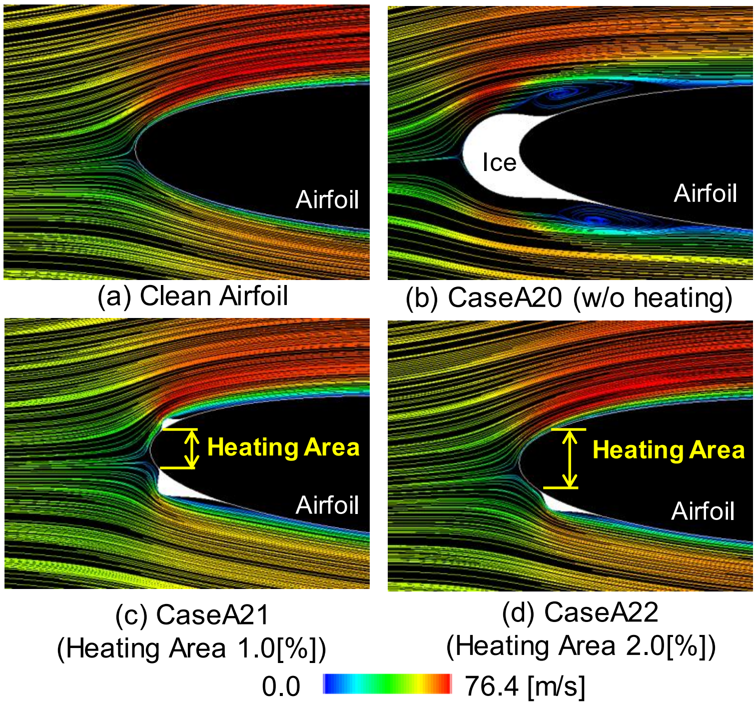

Figure 2 Streamline and ice shape of the airfoil. White regions represent adhered ice shape, and the color of the streamlines denotes the flow speed.

3 RESULT AND DISCUSSION

Figure 2 shows the accreted ice shapes and streamlines for cases consisting of clean, A20, A21, and A22. The color of the streamlines denotes the flow speed. In Fig. 2 (b), the large ice is formed around the leading edge, and recirculation regions appear behind the ice. Owing to add the heater, as shown in Fig. 2 (c) and (d), the ice at the leading edge of the wing vanished, though the small ice remains at the downstream of heater. However, the recirculation regions disappear in applying the heater. Obviously, ice formation strongly depends on the heating area and the attack angle.

Figure 3 indicates lift coefficient C_l and normalized drag to represent the aerodynamic performance. Without heating, C_l increase non-linearly with the increase of the attack angle. By applying the heater, C_l drastically recovers and increases almost linearly with the increase of the attack angle. Therefore, the heater works well to prevent a lift loss even in applying 2.0% chord length. On the other hand, the drag distributes more complexly. Interestingly, in case that the attack angle is 0.0 degree, the drag increases by setting the 1.0% heater in comparison to the without heating case. Then, the drag decreases by applying the 2.0% heater. In the cases that the attack angle is 2.0 degrees, the drag decreases with expanding heating area. However, in case that the attack angle is 4.0 degrees, the drag increases with expanding heating area. Clearly, the drag distribution changes irregularly by the heating area.

Figure 4 shows the heating power ratio normalized by the thrust power. The heating power ratio decreases by increasing the attack angle and heating area. It is confirmed that the heating ratio is less than 0.4%, therefore, the heating power is extremely lower than the trust power.

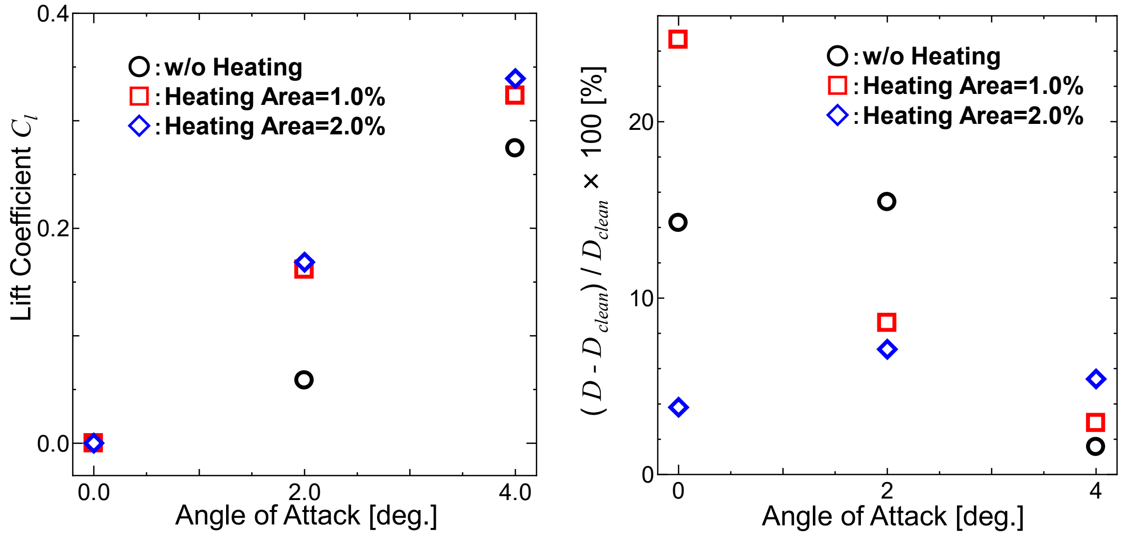

Figure 3: Lift coefficient (left) and drag change from the clean airfoil (right). The drag is normalized by the drag for clean airfoil at the same attack angle.

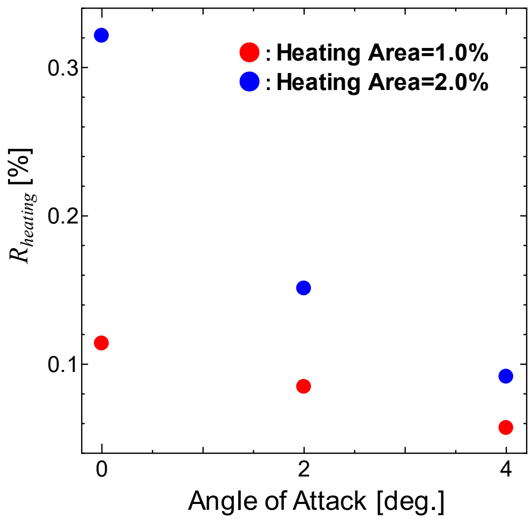

Figure 4: Heating power ratio against thrust power.

4 CONCLUDING REMARKS

The icing simulation of a NACA0012 airfoil is performed to investigate the effect of the surface heating area on the icing and the aerodynamic performance. The Extended Messinger model for the thermodynamics calculation is modified to simulate the heated surface. As a result, the findings obtained in this study are as follows.

- Ice formation strongly depends on the heating area and the attack angle of the airfoil.
- In the unheated case, the large ice forms around the leading edge and induces the flow separation behind the ice. It results in the increase of the fluid drag.
- The heater applied at the leading edge prevents the ice formation, and it significantly improves the aerodynamics performance, i.e., increase of lift coefficient.
- For the case that attack angle is 0.0 degree, the drag coefficient increases at the case of 1.0% heating area, while it decreases at the case of 2.0% heating area.
- The heating power is very small comparing the thrust power.

REFERENCES

[1] Al-Khalil K. M. et al., "Validation of NASA thermal ice protection computer codes part 3: the validation of ANTICE", *NASA TM-2001-210907*, (2001), 12 pp.

[2] Bu X. et al., "Numerical simulation of an airfoil electrothermal anti-icing system", *Aerospace Eng*. Vol. 227, No. 10, (2013), pp. 1608-1622.

[3] Reid T. et al., "FENSAP-ICE: unsteady conjugate heat transfer simulation of electrothermal de-icing". *J. Aircraft*, 49(4), (2012), pp. 1101-1109.

[4] Kato M. and Launder B. E., "The modeling of turbulent flow around stationary and vibrating square cylinder", *Proc. 9th Symp. Turbul. Shear Flows*, (1993), pp. 10-4-1-10-4-6.

[5] Schiller L. and Naumann A., "A drag coefficient correlation", *Z. Ver. Deutsch*, Vol. 77, (1935), pp. 318-320.

[6] Özgen S. and Canibek M., "Ice accretion simulation on multi-element airfoils using extended Messinger model", *J. Heat Mass Transfer*, Vol. 45, No. 3 (2009), pp. 305-322.

[7] Olsen W. et al., "Ice shapes and the resulting drag increase for a NACA 0012 airfoil", *NASA-TM-83556*, E-1935, (1994), 30 pp.